

STRUCTURAL ANALYSIS OF PROPULSION SYSTEM COMPONENTS OF AN INDIGENOUS CRYOGENIC ROCKET ENGINE

A.K.Asraff^{*}, R.Muthukumar[†], T.J.Ramnathan[‡], C.G.Balan[§]
Liquid Propulsion Systems Centre, ISRO, Valiamala
Thiruvananthapuram- 695 547, INDIA.

This paper deals with structural analysis of major liquid propulsion system components of an indigenously developed cryogenic rocket engine viz. thrust chamber, gas generator, fuel turbo-pump, and oxygen turbo-pump. Stress analysis of ground facilities for hot testing of GG and TP's are also discussed. Material, geometric and contact nonlinearities are considered in the analysis. Double walled regions of the thrust chamber are idealized using layered shell elements whereas for other zones, uni-layered shell elements are employed. Ground test facility ducts, supports, bellows, flexible hoses etc. are idealized using pipe, beam and spring elements. Both thermal and pressure loads are accounted for in the analysis. High temperature mechanical properties of the engine materials available from in-house testing as well as literature are employed for analysis.

I. Introduction

The Indian Space Research Organisation has embarked upon the design and development of a cryogenic rocket engine which works on LOX/LH2 propellants. This engine is based on the gas generator cycle and is intended for the Mark III version of Geosynchronous Satellite Launch Vehicle. The engine is to develop a thrust of 200 kN in vacuum. It consists of various subsystems like thrust chamber, gas generator, turbo-pumps etc. Structural analysis of these pose a challenging task since (a) the effect of thermal and pressure loads have to be considered simultaneously (b) various load conditions like bolt pre-torquing, chill down trial, hot test conditions etc. have to be analyzed (c) structural integrity has to be verified and ensured for ground test as well as flight configurations and (d) effect of the mounting structures/connecting ducts have also to be considered. This paper presents the details of structural analysis carried out using the finite element technique to evaluate their structural integrity before conducting ground hot tests.

II. Stress analysis of gas generator & oxygen turbo-pump test facility

Figure 1 shows the details of the ground hot test facility for the gas generator of the engine. In the flight engine, hot gases emanating from the GG first passes through a bend pipe and drives the LH2 turbopump. Subsequently, they are admitted to the LOX turbopump before dumping overboard. Configuration of the GG & bend pipe are shown in figure 2. The GG is of double walled construction which is regeneratively cooled and fabricated out of stainless steel whereas the bend pipe is of Inconel alloy. Layout of the GG hot test facility is shown in figure 2. In this configuration, the hot gases from GG first passes through the bend pipe and are then admitted to the ground facility ducts. It subsequently passes through a gas cooler before letting out to the atmosphere through a long disposal duct. Flexible hoses and axial bellows are provided in hot gas ducts of the ground test facility to take care of thermal expansion.

All the pipe lines are adequately supported both vertically to bear its self weight and horizontally to act as anchors where bellows are provided. This is to bear the pressure induced end thrusts on the pipe segments. The gas cooler is meant to reduce the temperature of the hot gases by about 300 K thus simulating the thermal mass of the LH2 TP. This cooled gases can subsequently be used to test the OTP, if required. The gas cooler is a cylindrical shaped heat exchanger in which the hot gases coming from the GG are cooled by passing it through a number of coolant tubes which are continuously cooled on its outer surface by circulating water.

Finite element modeling of the whole ducting starting from the GG to the end of the disposal duct is done using one dimensional pipe elements. The supports and anchors for these pipes are also modelled based on their dimensions in order to incorporate their true stiffnesses instead of assuming simplistic boundary

^{*} Deputy Head, Structural Dynamics & Design Analysis Division, akasraff@yahoo.com

[†] Head, Structural Dynamics & Design Analysis Division, r_muth_kumar@rediffmail.com

[‡] Group Director, Mechanical Design & Analysis Group, tjrkv@yahoo.com

[§] Deputy Director, Liquid Propulsion Engines

conditions. Ducts are idealized using PIPE16 elements which have thermal and pressure loading capability. Structural analysis has been carried out using ANSYS (Revision 11) FEA package (Ref.1) implemented in a Windows based high performance workstation.

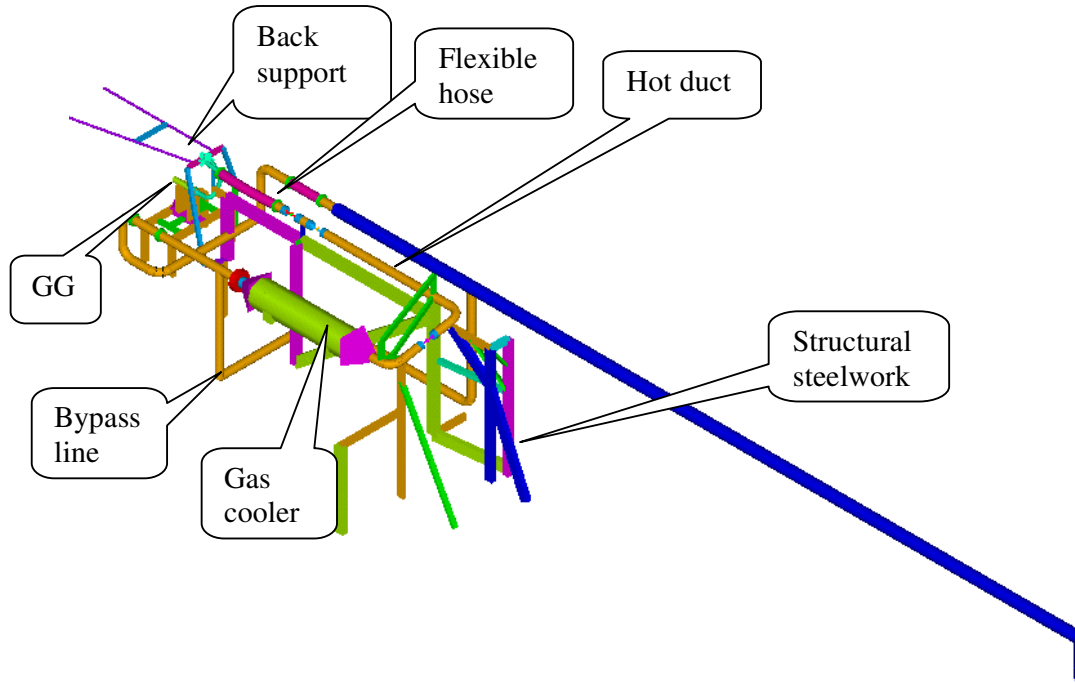


Figure 1. Configuration of the GG hot test facility

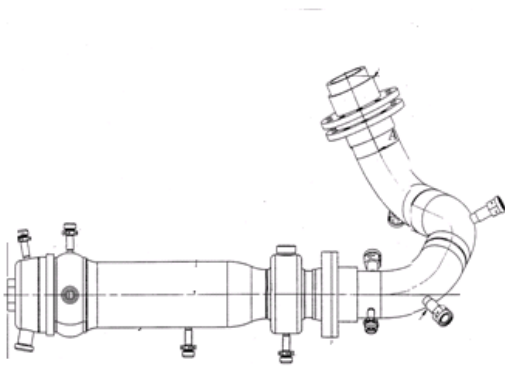


Figure.2. Configuration of gas generator & bend pipe

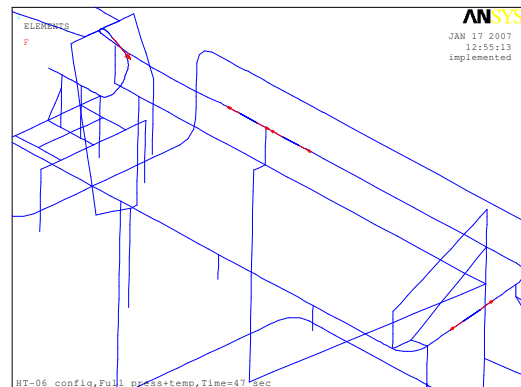


Figure3. Finite element model of GG test facility ducts

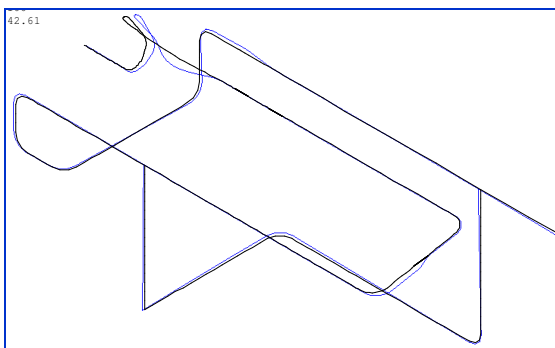


Figure 4. Deformed shape of GG test facility ducts

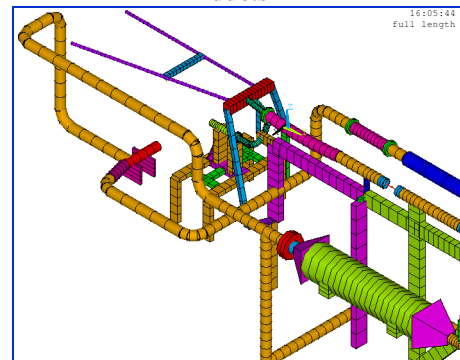


Figure 5. Configuration of OTP test facility ducts

The supporting and anchoring structural steel members are idealized using BEAM4 elements. Support for the GG test article is also modeled using the above elements. Bellows are modeled using COMBIN14 elements. This is a spring-damper element capable of representing longitudinal or torsional springs. Each bellows is idealized using 6 such elements, 3 elements representing the axial stiffness in nodal X, Y and Z directions whereas the other 3 elements for the rotational stiffness about nodal X, Y and Z axes. Axial stiffness of the bellows furnished by the manufacturer is used to evaluate all other stiffnesses based on the bellow convolution geometry (Ref.2) Compression load testing of one of the bellows is also conducted to verify its axial stiffness. Base of all supporting structural steel members is assumed fully clamped since it is concreted to the ground. A back support is provided to the bend pipe to bear the pressure thrust load. The back support horizontal plate is idealized using SHELL43 elements. Node to surface gap elements are defined between the 1 dimensional elements used for idealizing the bend pipe and the shell elements of the back support. Figure 3 shows the finite element model employed for analysis. The pressure separating forces in each bellows are applied on either end so as to keep them in tension when internal pressure is applied. Flexible hoses are idealized as thin walled isotropic pipes using PIPE16 elements with reduced Young's modulus.

Deformed shape of the hot gas ducts is shown in figure 4. The flexible hose is found to flex considerably in the lateral direction and absorb thermal expansion of the GG and bend pipe. Actual hot testing also showed the same behaviour. After a few initial trials, the hot test could be conducted successfully without any leak or structural failure.

After successful completion of the GG hot testing, the facility is modified as given in figure 5 in order to hot test the oxygen turbopump. Stress analysis of the modified facility has also been done to confirm the structural integrity of the OTP as well as the facility ducts.

III. Stress analysis of oxygen turbopump

Configuration of the oxygen TP is shown in figure 6. The casing of the OTP is fabricated out of stainless steel and the rotor is of Inconel. Axisymmetric stress analysis of the OTP casing and rotor conducted separately (Ref.3) for flight hot run condition showed positive margin against yielding. Stress analysis of the OTP and mounting fixtures for ground based hot test condition is described here. The turbopump is placed on two vertical saddle shaped mounting plates of the support fixture, one each on the pump and turbine sides. The supporting fixture is in turn bolted to a massive steel platform. Eight M8 bolts of A-286 make are used to fasten the pump side flange to the mounting plate whereas five M8 bolts of Inconel make are used on attach the turbine side flange. The base plate is bolted to the facility plate using four M16 Unbrako bolts, two each on the pump and turbine sides. The turbine side plate is L shaped and is bolted to the base plate through slotted holes which can permit thermal expansion/contraction.

Turbopump and fixture plates are idealized using SHELL181 elements whereas the bolts are modelled using BEAM4 elements. Only 180° portion of the TP and support plates are modelled to exploit symmetry. The rotor is not considered in the model since casing alone participates with the test facility in bearing loads. Figure 7 shows the details of the finite element model employed for analysis. Surface to surface contact elements are defined between the flanges on the TP and the corresponding mounting plates to simulate stresses due to pre-torquing of the bolts. The contact surfaces are offset corresponding to the gap between the contact and target elements so as to model the thickness of the mating flanges.

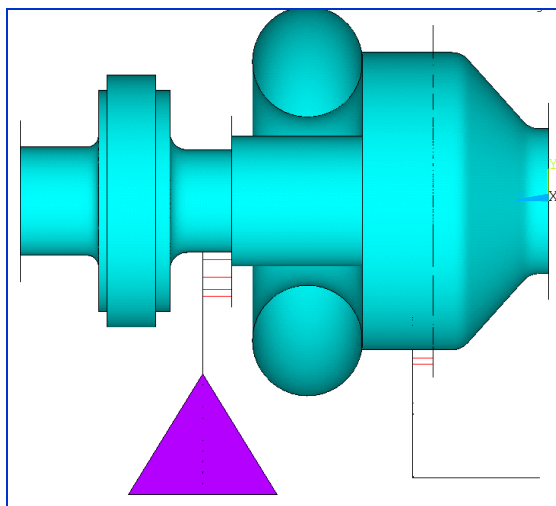


Figure 6. Configuration of oxygen turbopump

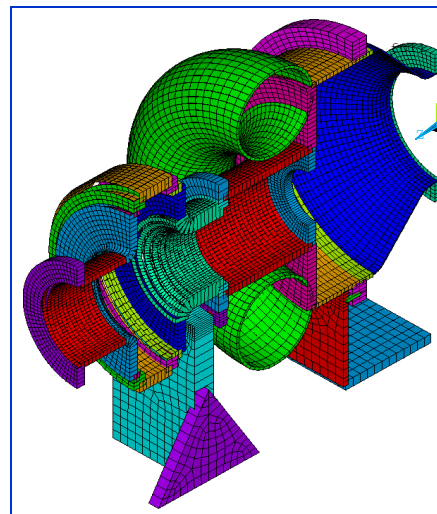


Figure 7. Finite Element model of OTP

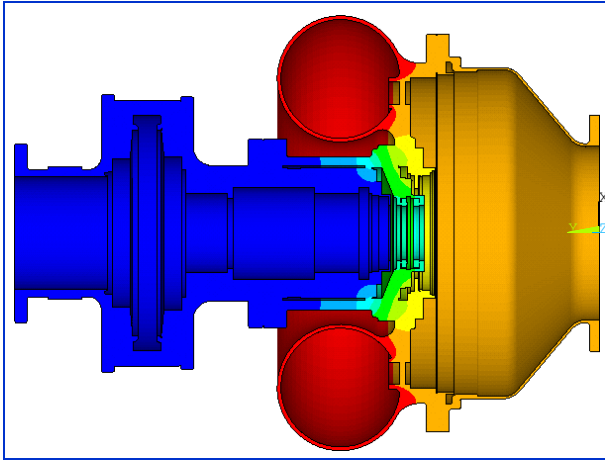


Figure 8. Temperature distribution in OTP for hot test

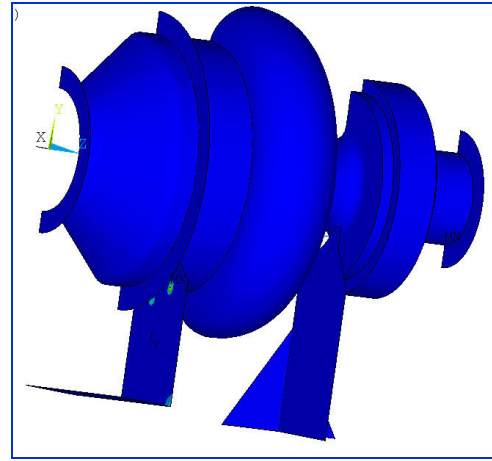


Figure 9. Plastic strains in OTP

Combined material and geometric nonlinear stress analysis is carried out using Bilinear Kinematic hardening plasticity model for the turbopump materials. Stress analysis is conducted in two load steps: In the first load step, the only loading applied is the bolt pre-torque to the level of 40% of yield strength of bolt material for the mounting bolts. In the second load step, pressure and thermal loads are imposed on the above model. During chilling operation, liquid nitrogen at 77 K will be circulated on the pump side whereas the turbine side will be at room temperature. The turbopump is found to tilt as a whole with the pump tip dipping down due to thermal contraction.

For the hot test condition, the pump side will experience low temperature of 77 K whereas the turbine side will be in contact with hot GG gases at high temperature. Temperature distribution in the turbopump TP is shown in figure 8. In this case also, the TP is found to tilt globally. Yielding to the tune of 3.6% plastic strain is observed in a local area around the bolt holes in the turbine side mounting plate as shown in figure 9. This is ignored since (a) it is partially due to the idealization of the bolt as a beam element and (b) the material of the mounting plate is a ductile variety of stainless steel which can take high amount of yielding before fracture. Bolt stresses are within the allowable limits.

Hot testing of the OTP could be completed successfully in the first trial itself for the rated duration without any leak, opening up of the bolted connections or structural failure anywhere.

IV. Stress analysis of fuel turbopump

The fuel turbopump casing is also fabricated out of stainless steel and rotor shaft, disk and blades out of Inconel. Configuration of the fuel TP and mounting fixture plates are shown in figure 10. Detailed stress analysis of the FTP (Ref.4) for flight hot run condition showed sufficient margins against yielding. Stress analysis for ground hot test condition is reported here. The turbopump and fixture plates are idealized using SHELL143 elements whereas the mounting bolts are modelled using BEAM4 elements. Figure 11 shows the details of the finite element model employed for analysis.

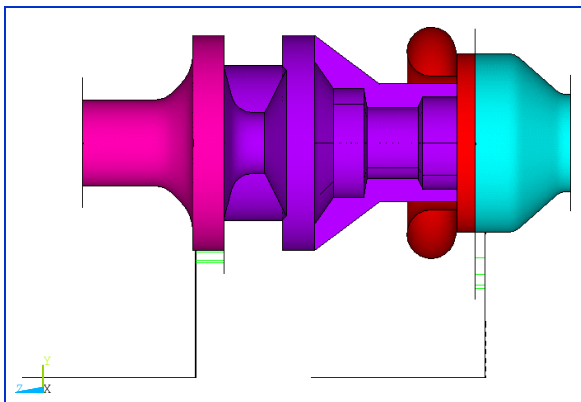


Figure 10. Configuration of fuel turbopump

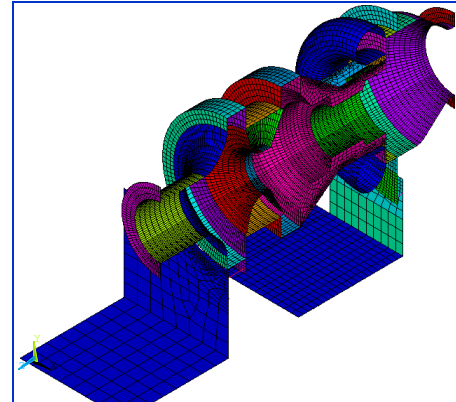


Figure 11. Finite Element model of FTP

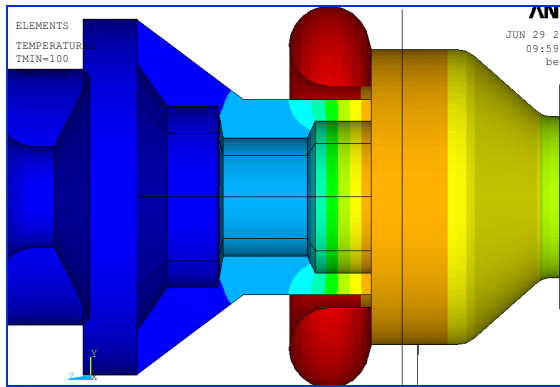


Figure 12. Temperature distribution in FTP for hot test

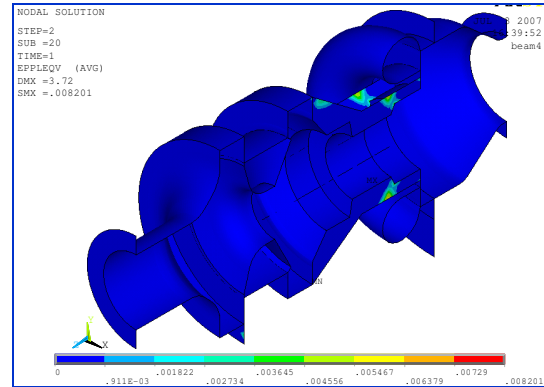


Figure 13. Plastic strains in FTP

For the hot test condition, the pump side will be in contact with liquid nitrogen and the turbine side will experience hot gases from the gas generator. Steady state temperature distribution in the TP for the above test condition is shown in figure 12. Stress analysis results indicate that the TP is found to tilt globally in this case also due to pump contraction. The casing fins on the turbine side are found to yield, the maximum plastic strain being 0.8% in the underside fin as shown in figure 13. This is not considered to be detrimental to the structural integrity of the turbopump. Actual hot testing of the FTP is yet to commence.

V. Structural analysis of thrust chamber

The thrust chamber of this engine is of double walled construction with a nozzle exit area ratio of 100. A high conductivity copper alloy is selected for the inner wall in regions of high heat flux whereas for other regions a special grade stainless steel is chosen. The outer wall is mostly of stainless steel. Regenerative cooling of the thrust chamber is achieved by passing liquid hydrogen through rectangular coolant passages milled on outer surface of the inner wall. The coolant channels are of helical pattern up to an area ratio of 10 after which they are kept straight. The number of coolant passages also varies along the length of the chamber. The outer and inner walls are bonded together by brazing at high temperature. The convergent and divergent subassemblies of this chamber are separately fabricated by brazing together the respective inner and outer walls. These subassemblies are subsequently joined together at the throat by welding the convergent and divergent inner walls initially and then the corresponding outer walls. A longitudinal section of the thrust chamber is shown in figure 14. Thermo-structural analysis & cyclic life prediction of the thrust chamber using an axisymmetric FE model showed adequate cyclic life of 48 hot runs (Ref.5). Stress analysis and optimization of the injector head assembly is also conducted to verify its adequacy against hot test and ground based acceptance pressure test conditions (Ref.6).

Structural analysis of the engine while undergoing hot tests at the High Altitude Test facility is dealt with here. Three sets of horizontal spacers made of stainless steel are provided to mount the engine to a rigid wall. A thrust measurement system is also incorporated in one of these spacers to measure thrust accurately during a high altitude test. The engine will be mounted inside a vacuum chamber for testing. The spacers are adequately anchored to the ground. Since the thrust chamber possesses symmetry about one axis, only 180° portion is modeled as shown in figure 15. The thrust chamber is modelled using SHELL181 element available in the ANSYS element library. This relatively newer element can model shell structures using sections (layers) having different thicknesses and material properties. The double walled portion of the chamber is idealized using 3 sections, the first section representing the inner wall, the 2nd section simulating the ribs and the 3rd section for the outer wall. The ribs are idealized by assigning orthotropic properties for the 2nd section. Helix angle of the ribs is also accounted for.

Stress analysis of the thrust chamber has been carried out for hot test conditions by applying gas pressure and temperature corresponding to a thrust level of 200 kN. Figures 16 and 17 show the distribution of gas pressure and wall surface temperatures respectively. A load factor of 1.1 is applied for pressure loading and temperatures are incremented by 28 K for conducting stress analysis. Weight of the total engine is simulated by modeling mass elements at the CG of the engine and applying acceleration loading of 1g in the upward direction. The upstream end of the first spacer assembly are arrested in all directions to simulate the fixity of the rigid wall of the HAT facility. The engine system is adequately anchored to the ground as shown in figure 18. Figure 19 shows the deformed shape of the thrust chamber due to thermal, pressure and self weight of the engine.

Free vibration analysis of the engine assembly is also carried out to evaluate its natural frequencies and mode shapes, for a subsequent random vibration analysis. A full 360° FE model of the thrust chamber is employed in this case. No thermal or pressure loads are considered. The first mode is found to be a cantilever mode of the thrust chamber with a natural frequency of 21 Hz as shown in figure 20.

Random vibration analysis of the engine assembly is also carried out in order to evaluate the vibration induced deflection of the chamber nozzle during hot testing of the engine. A trial base excitation in the vertical direction is specified as a flat spectrum with a maximum Power Spectral Density value of $0.03 \text{ g}^2/\text{Hz}$ from 20 to 2000 Hz. The chamber tip is found to deflect laterally by 33 mm (3 sigma value) for the above input level as shown in figure 21. This result is used to decide the radial clearance required between the chamber tip and the diffuser duct inlet so that during a hot test, the chamber tip would not hit the HAT facility ducts.

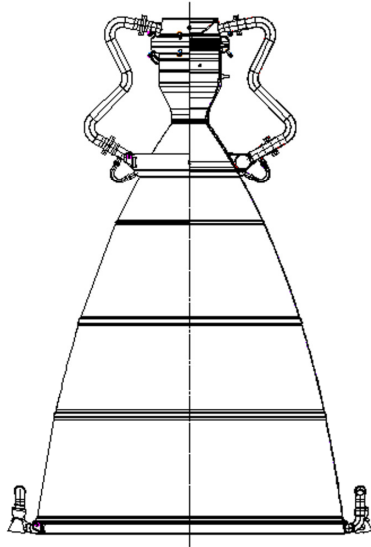


Figure 14. Configuration of thrust chamber

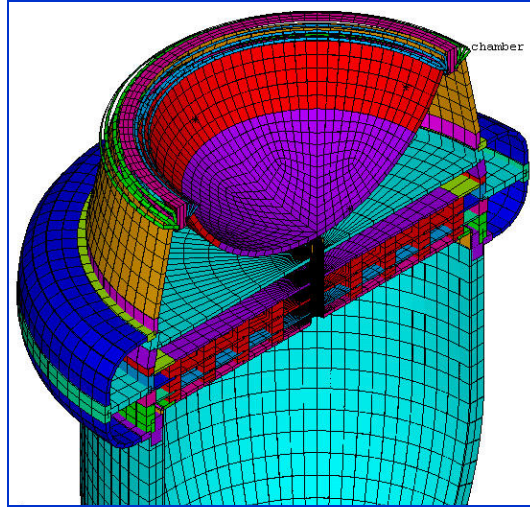


Figure 15. Finite element model of thrust chamber

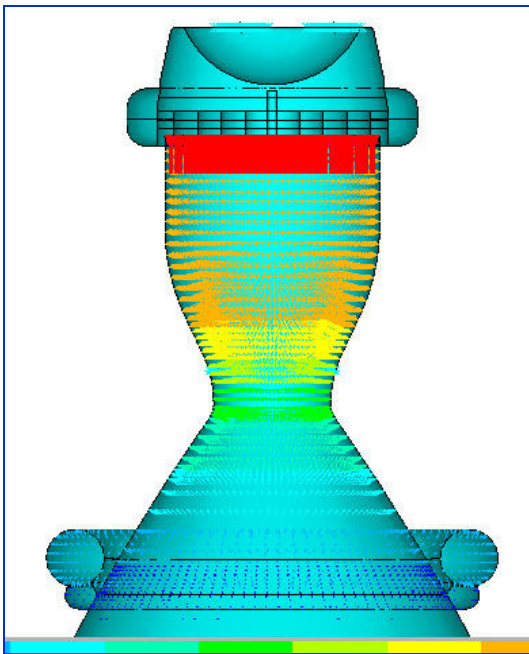


Figure 16. Pressure distribution in thrust chamber

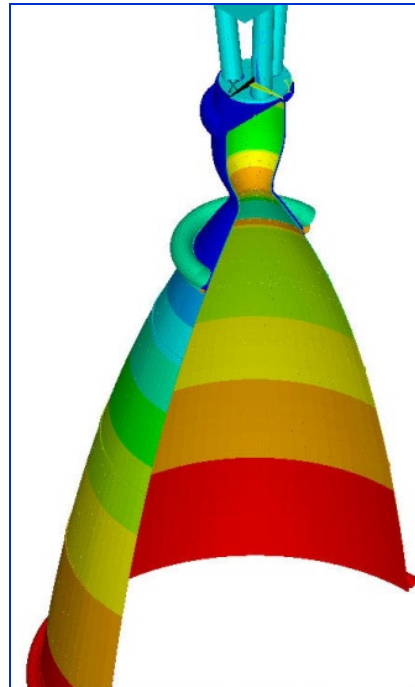


Figure 17. Thermal distribution in thrust chamber

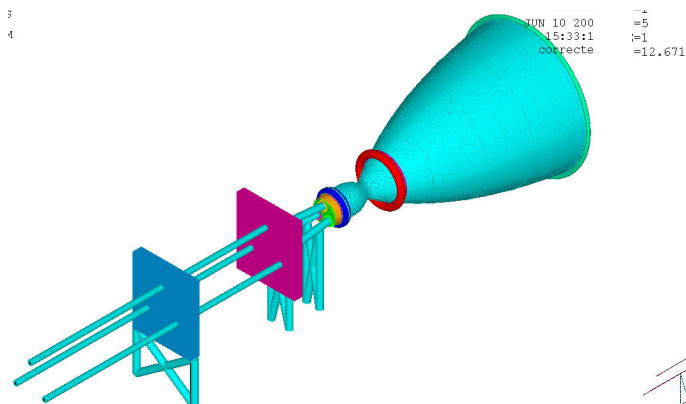


Figure 18. Configuration of supports at HAT facility

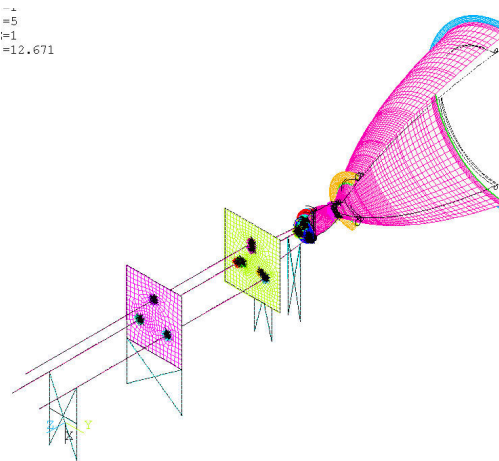


Figure 19. Static deflected shape

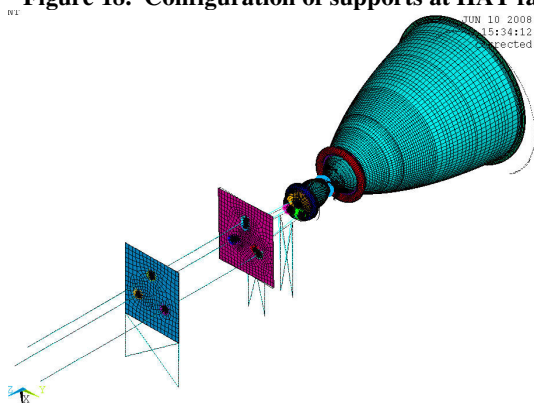


Figure 20. First vibration mode shape

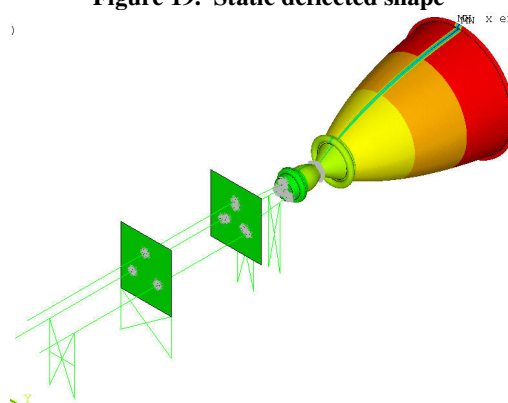


Figure 21. Dynamic deflection contours

VI. Conclusion

Structural analysis of some of the major subsystems of an indigenously designed cryogenic rocket engine is presented in this paper. Various novel techniques are incorporated in modeling and analysis of the same. Hot testing of some of the subsystems have been completed and others are in an advanced stage of being tested. Supports and anchors for the facility ducts and plates could be designed based on the results from FEA. Prediction of deflections and stresses from analysis have matched excellently with test results, validating this valuable tool for design verification.

Acknowledgments

A.K.Asraff thanks the excellent support extended by Mr.V.Viswanath of SDDAD for structural analysis and the encouragement, support and data related to propulsion furnished by Dr.V.Narayanan and his team of LPSC/ISRO.

References

- ¹"ANSYS – Engineering analysis system", Version 11.0, ANSYS Inc, USA, 2007.
- ²Dieter K.Huzel & David H.Huang, "*Design of Liquid Propellant Rocket Engines*", NASA SP-125, 1967, National Aeronautics & Space Administration, USA.
- ³A.K.Asraff, "*Revised stress analysis of CE-20 LOX turbine disk and casing*", Report No. LPSC/MDAG/SDDAD/TR/21/2006, August 2006 (unpublished).
- ⁴A.K.Asraff, "*Stress analysis of CE-20 LH2 turbine disk*", Report No. LPSC/MDAG/SDDAD/TR/04/2006, June 2006 (unpublished).
- ⁵A.K.Asraff, S.Sunil, R.Muthukumar, T.J.Ramanathan, "*New Concepts in Structural Analysis & Design of Double Walled LPRE Thrust Chambers*", AIAA Paper No. 2006-4368, 42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, July 2006.
- ⁶A.K.Asraff, "*Stress analysis & re-design of CE-20 thrust chamber*", Report No. LPSC/MDAG/SDDAD/TR/15/2006, December 2006 (unpublished).